

NAGW-478 1
7N-34-CR
270024
13P

ADAPTIVE FINITE ELEMENT FLUX
CORRECTED TRANSPORT TECHNIQUES
FOR CFD

K. Morgan, J. Peraire and R. Löhner	
I.N.M.E.	Code 4040
University of Wales	Naval Research Laboratory
Swansea SA2 8PP	Washington, DC 20370
United Kingdom	U.S.A.

INTRODUCTION

In previous papers [1,2] we have described an explicit finite element solution procedure for the compressible Euler and Navier-Stokes equations. The approach was a finite element equivalent of a two-step Lax-Wendroff scheme and was implemented on unstructured triangular or tetrahedral grids. An important feature of the work was the use of adaptive mesh refinement methods for the solution of steady state problems in 2D, using error indicators based upon interpolation theory.

In this paper, some recent developments in the extension of this approach are considered. We will describe how the basic solution procedure can be modified in a straightforward manner to produce a high resolution scheme on unstructured grids. This is accomplished by utilizing, in a finite element context, Zalesak's [3] multidimensional extension of the flux corrected transport (FCT) ideas of Boris and Book [4]. The problem of triangular mesh generation will be addressed and the adaptive mesh approach will be widened to handle 2D problems involving strongly transient phenomena. This will be implemented by allowing adaptive refinement and derefinement of the mesh as the solution proceeds. Finally, it will be demonstrated how directional refinement procedures can be incorporated for the efficient computation of steady 2D flows involving significant 1D features.

BASIC ALGORITHM

The basic solution algorithm will be briefly described for the two dimensional Navier-Stokes equations written in the conservative form

$$\frac{\partial \underline{U}}{\partial t} + \frac{\partial \underline{F}_j}{\partial x_j} = \frac{\partial \underline{G}_j}{\partial x_j} \quad (j=1,2) \quad (1)$$

where \underline{U} is the vector of conservation variables and \underline{F}_j and \underline{G}_j denote the advective and viscous flux vectors respectively. A time-stepping scheme for this

equation can be developed, in an operator-split fashion, by treating the diffusion terms in an explicit manner and the advective terms in the Lax-Wendroff fashion [5]. The result is that

$$\underline{u}^{m+1} = \underline{u}^m - \Delta t \frac{\partial \underline{F}_j^m}{\partial x_j} + \frac{\Delta t^2}{2} \frac{\partial}{\partial x_j} \left[\underline{A}_j^m \frac{\partial \underline{F}_k^m}{\partial x_k} \right] + \Delta t \frac{\partial \underline{G}_j^m}{\partial x_j} \quad (2)$$

where a superscript m denotes an evaluation at time $t = t_m$, $t_{m+1} = t_m + \Delta t$ and

$$\underline{A}_j = \frac{d \underline{F}_j}{d \underline{U}} \quad (3)$$

The spatial domain, Ω , is discretized using 3-noded linear triangular elements and a weighted residual [6] form of equation (2) is considered. The resulting integrals are evaluated exactly (in a 2-step fashion by firstly calculating an element level approximation to $\underline{u}^{m+1/2}$ [7]), leading to an equation

$$\underline{M} \underline{\delta U}^H = \underline{f}^m \quad (4)$$

where \underline{M} is the consistent mass matrix, $\underline{\delta U}^H$ is the vector of changes in the nodal values of \underline{U} over the timestep and the superscript H is introduced for use later. For the simulation of transient flows, this equation system can be solved iteratively and explicitly [8] and the method coupled with a domain splitting technique [9] to produce an efficient computational procedure. For steady flows, local time steps are employed and equation (4) is replaced by the explicit scheme

$$\underline{M}_\lambda \underline{\delta U}^H = \underline{f}^m \quad (5)$$

where \underline{M}_λ is the lumped diagonal mass matrix. It should be noted that, in 1D equation (5) reduces to the well-known finite difference scheme of Burstein [10].

For problems involving strong shocks, the solution \underline{u}^{m+1} is smoothed, by the application of artificial viscosity [11], before proceeding to the next time step.

FCT EXTENSION

A more robust solution scheme, giving better resolution of flow discontinuities, can be produced by adding an FCT procedure to the above process. Erlebacher [12] and Parrott and Christie [13] have demonstrated how Zalesak's [3] multidimensional extension of the FCT algorithm of Boris and Book [4] can be implemented on triangular grids. The idea is to combine a high order scheme with a low order scheme in such a way that the high order scheme is employed in regions where the flow variables vary smoothly, whereas the low order scheme is favored in those regions where the variables vary abruptly. The low order scheme

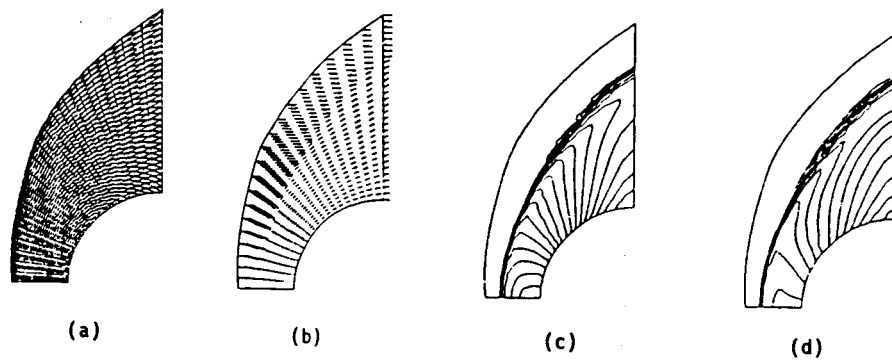


Figure 1

Mach 8 flow past a cylinder. (a) Mesh (b) Velocity vectors (c) Pressure contours (d) Density contours.

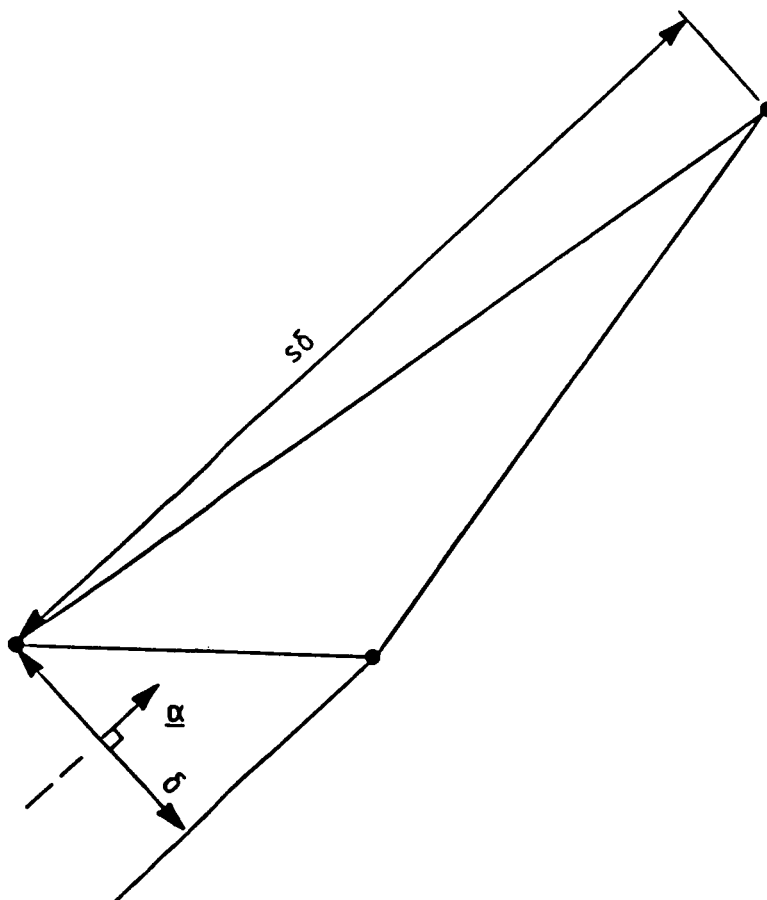


Figure 2

Definition of the mesh parameters δ , s and α .

4
should give monotonic results for the problem of interest.

The solution method of equation (4) will be used as a high order scheme and a high order solution, \underline{U}^H , after an time step can be defined by

$$\underline{U}^H = \underline{U}^m + \underline{\delta U}^H \quad (6)$$

Similarly, a low order solution, \underline{U}^L , is defined by

$$\underline{U}^L = \underline{U}^m + \underline{\delta U}^L \quad (7)$$

where the low order increment is calculated as

$$\underline{\delta U}^L = \underline{\delta U}^H + \underline{D} \quad (8)$$

and the smoothing term \underline{D} is given by

$$\underline{D} = C^L \underline{M}_\lambda^{-1} (\underline{M} - \underline{M}_\lambda) \underline{U}^m \quad (9)$$

where C^L is a constant. This form for the smoothing is suggested by the fact that at node i on a uniform grid in 1D

$$[\underline{M}_\lambda^{-1} (\underline{M} - \underline{M}_\lambda) \underline{U}^m]_i = (U_{i+1}^m - 2U_i^m + U_{i-1}^m)/6 \quad (10)$$

It should be noted that equations (6-8) can be re-arranged to give

$$\underline{U}^H = \underline{U}^L + \underline{D} \quad (11)$$

The new solution is computed according to

$$\underline{U}^{m+1} = \underline{U}^L + \underline{D}^* \quad (12)$$

where \underline{D}^* is obtained by writing the element contributions to \underline{D} , in such a way as to attempt to ensure that, \underline{U}^{m+1} is free from extrema not found in \underline{U}^m or \underline{U}^L [14, 15].

The numerical performance of this FCT scheme is illustrated in Figure 1 which shows the solutions obtained for the problem of Mach 8 flow past a cylinder.

Further generalizations of FCT are possible which offer interesting possibilities for future investigation [16].

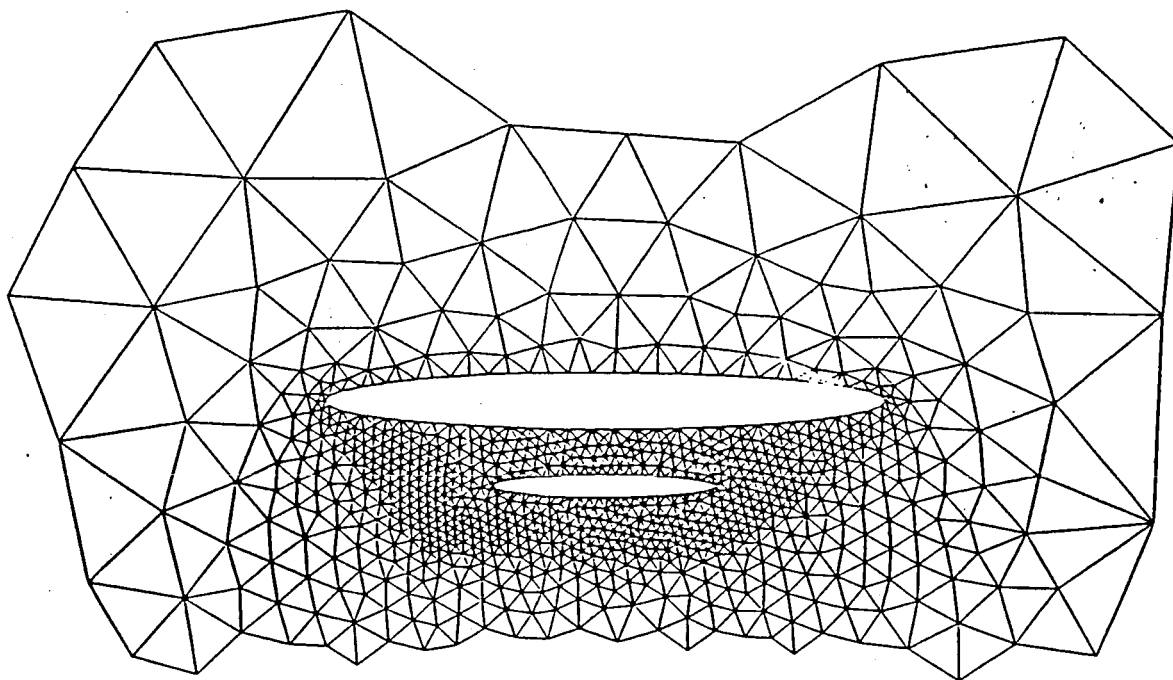


Figure 3

Detail of the initial mesh produced for the analysis of a store separation problem.

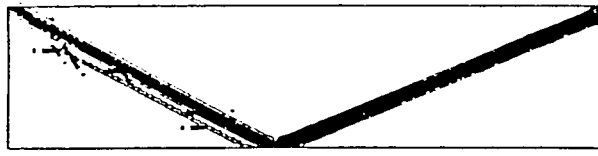
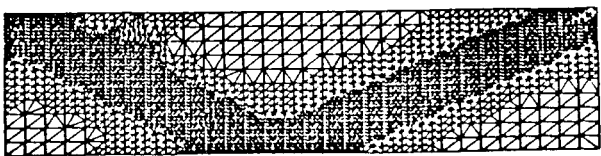
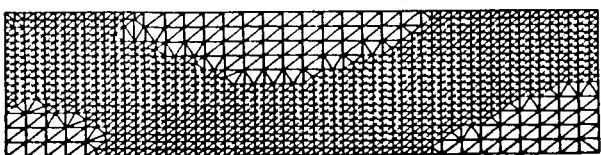
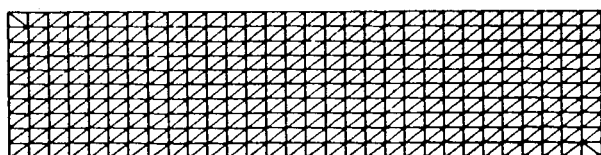


Figure 4

Regular shock reflection at a wall. Sequence of meshes produced using mesh enrichment and the corresponding pressure contours.

MESH GENERATION

The use of triangular elements in 2D means that computational domains of complex geometrical shape can be readily modelled and a variety of triangular mesh generation algorithms for planar domains are available [17, 18]. The approach to mesh generation to be outlined here begins by defining the boundaries of the solution domain in terms of Bezier polynomials and then covering this domain with a coarse 'background' grid of 3 noded linear triangles. This grid is normally constructed by hand and the only geometrical requirement imposed is that the solution domain should be completely covered by this grid i.e. the background grid is not required to approximate the geometry. At each node on the background grid, we specify the values of mesh parameters δ , s , α . During the mesh generation process, the local values of these parameters for the mesh being generated will be obtained by interpolation over the background grid. For the exact definition of these mesh parameters, it is useful to refer to Figure 2 which shows a typical generated triangle. The triangle has length $s\delta$ in the direction of α and length δ in the direction at right angles to α . We call δ the local node spacing, s the local degree of stretching and α the local direction of stretching. The full flexibility of the mesh generator need not be used to construct an initial mesh for a given problem, but it will be used in an adaptive mesh process to be described later. In particular, if a uniform distribution of δ is required, with no stretching, the background grid need only consist of a single element. The mesh generation process begins with the placing of the boundary nodes. The lines joining successive boundary nodes form the initial generation 'front', which is the collection of sides available to construct triangles. One side in the front is chosen, and a triangle is constructed with values of δ , s and α interpolated from the background grid. The front is updated and the process is repeated until the front is empty, at which stage the whole solution domain has been discretized. Full details of the mesh generation process can be found elsewhere [19].

The performance of the mesh generator is demonstrated in Figure 3, which shows a detail of the initial mesh produced for the analysis of a store separation problem. This problem has been used to demonstrate the full power of the generator by directly coupling it to the transient solution procedure and using it to locally regenerate the mesh as the store moves through the flow field [20].

ADAPTIVE MESH STRATEGIES

Adaptive mesh strategies have a major role to play in the development of efficient solution techniques for large problems in CFD. The ultimate objective is the ability to solve a given problem to a prescribed accuracy with the optimum number of grid points and, although this goal has not yet been met, major steps in this direction have already been made.

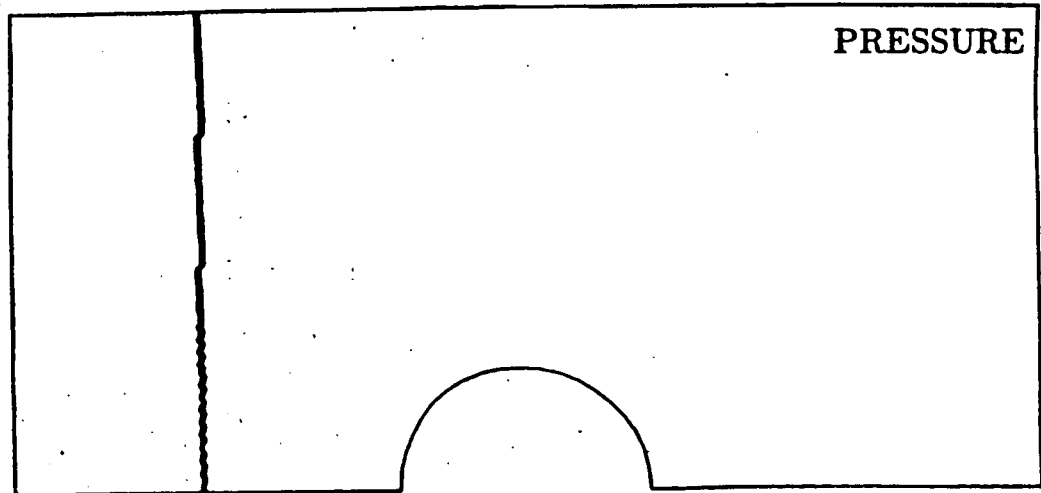
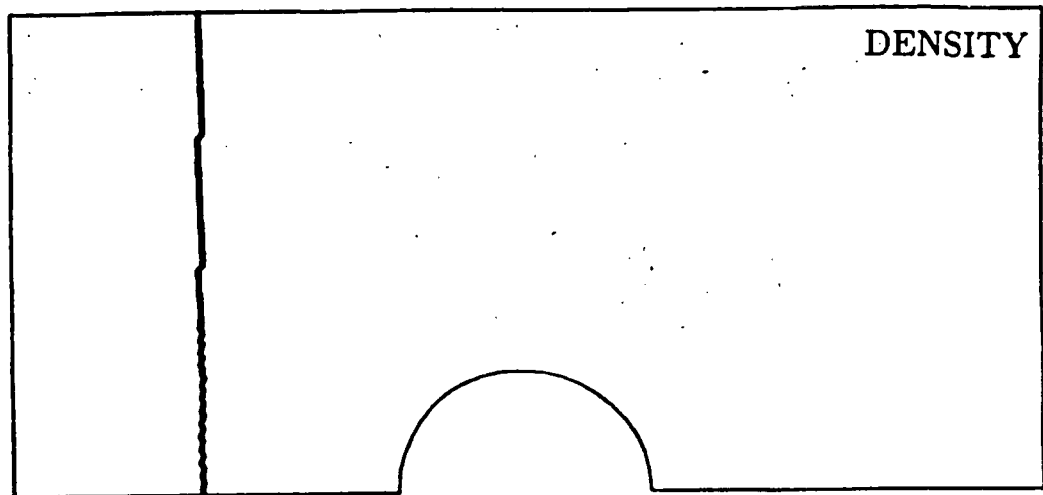
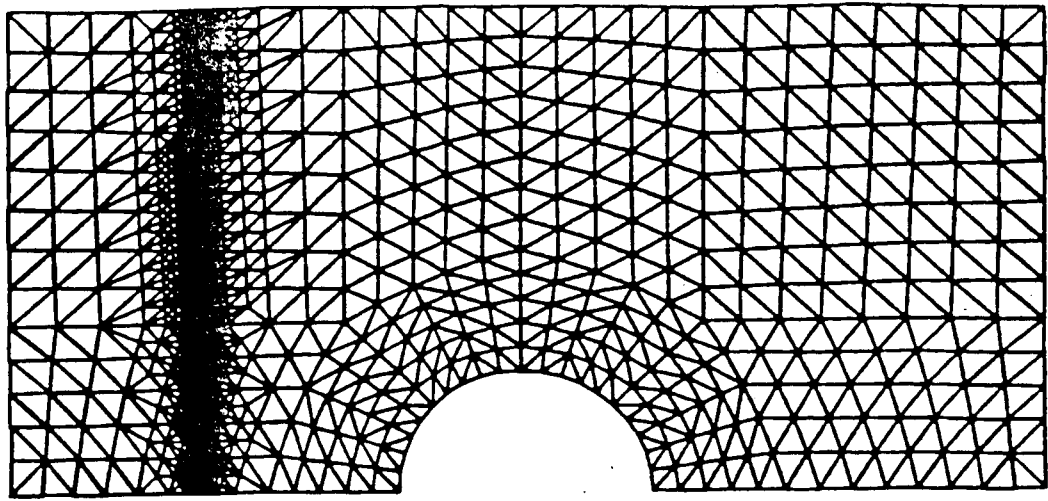


Figure 5

Shock impinging on a half-cylinder
(a) $t=0$, 2554 elements, 1335 points

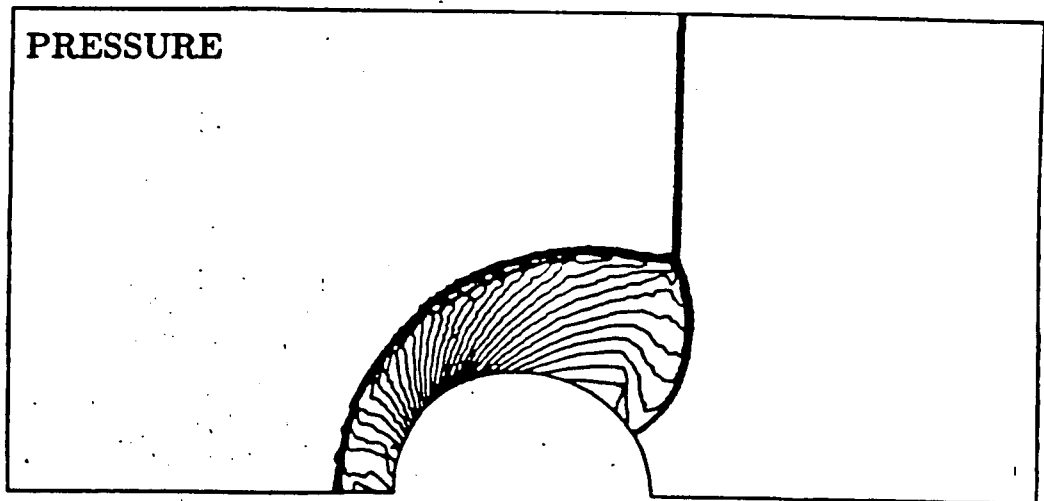
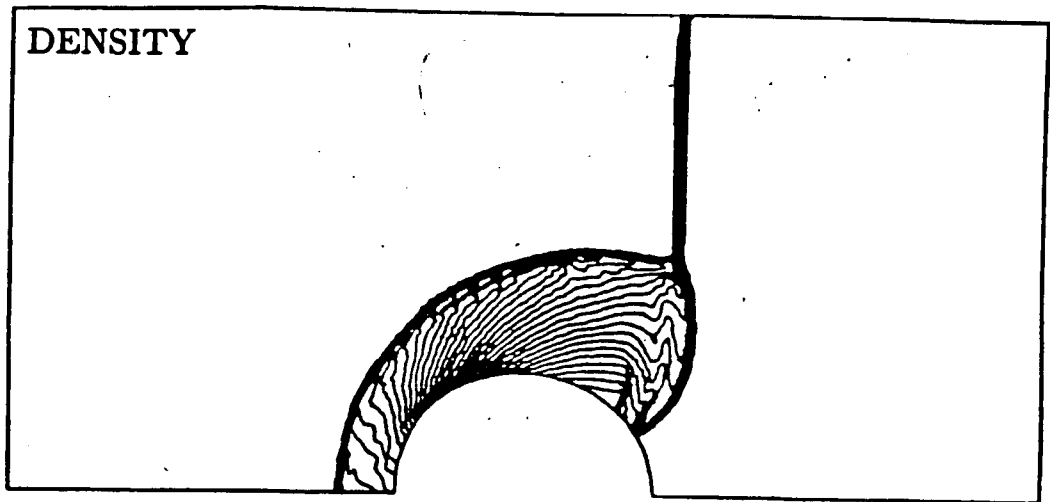
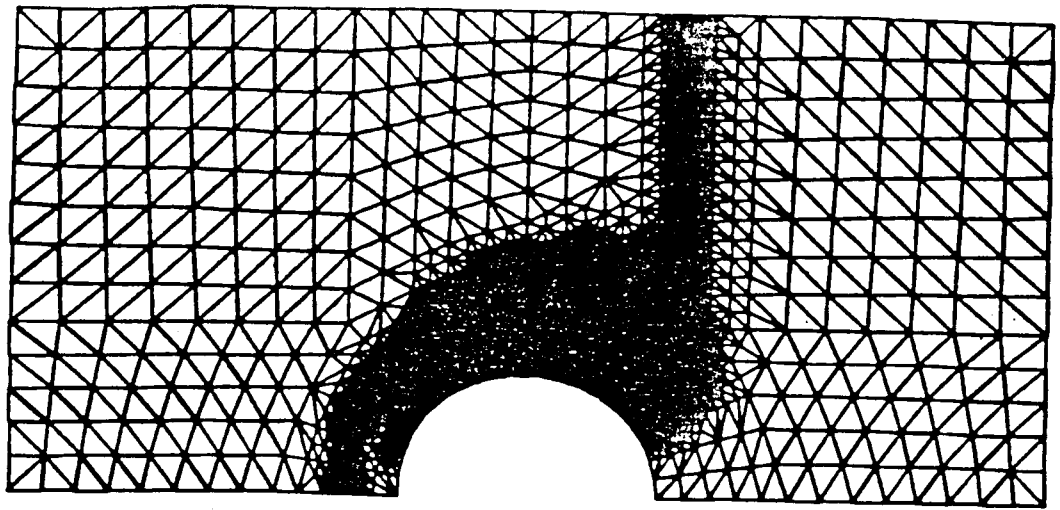


Figure 5 (cont)

(b) $t=0.3$, 9057 elements, 4626 points

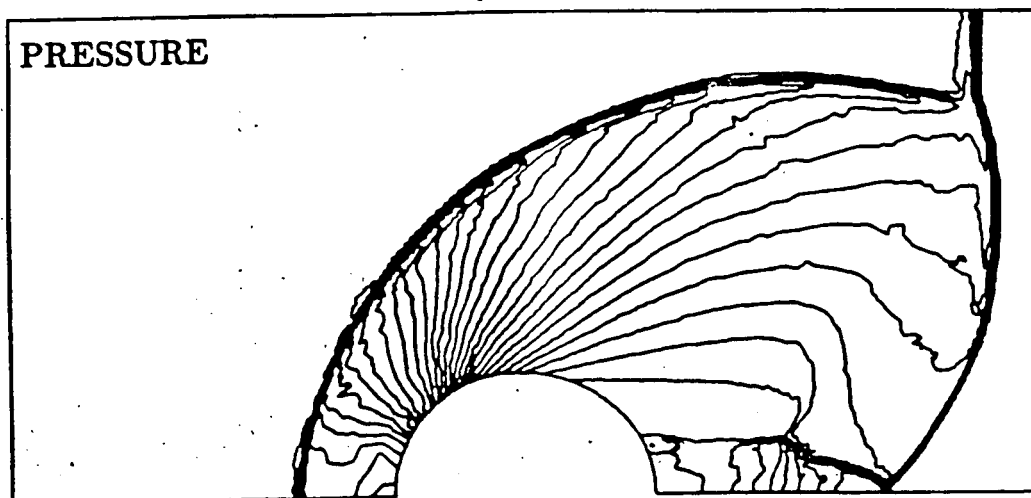
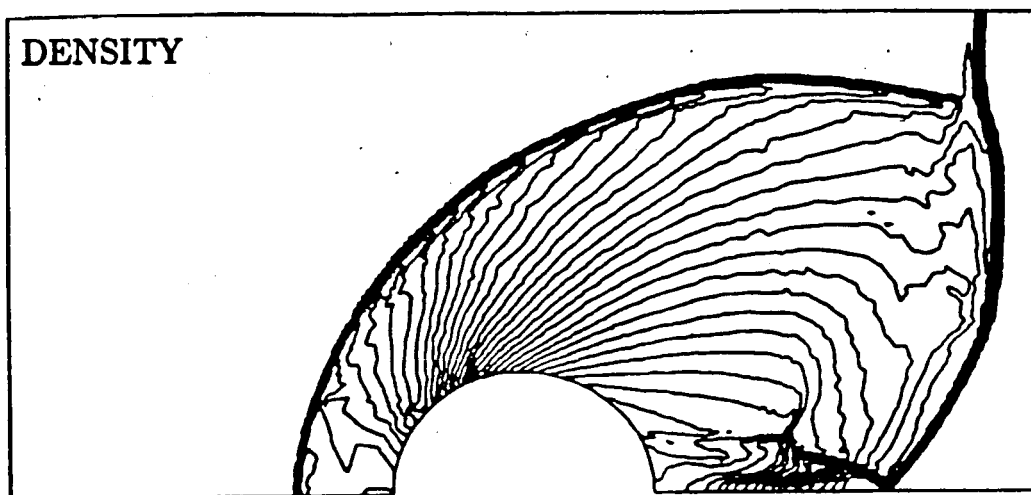
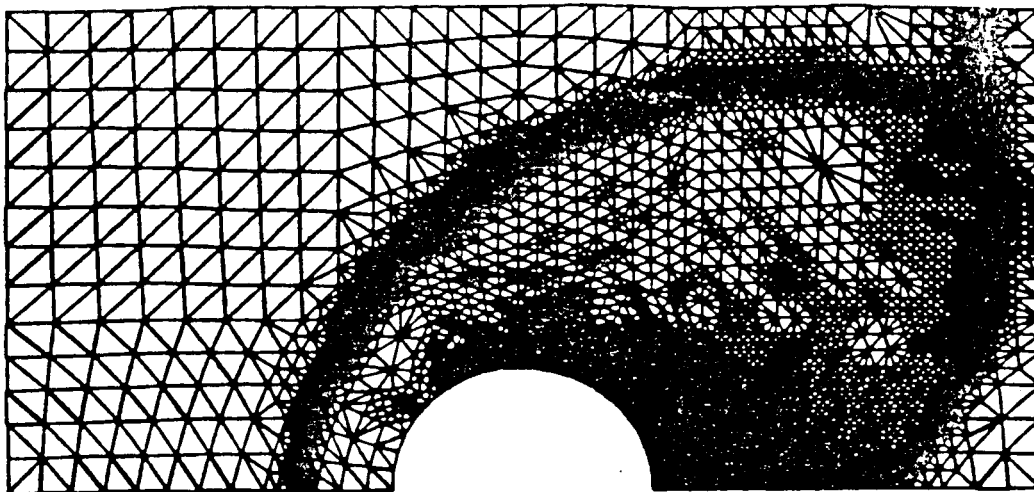


Figure 5 (cont)

(c) $t=0.5$, 12020 elements, 6129 points

Mesh movement technique can be contemplated [21, 22] but suffer from the drawback that the accuracy of the final computation can be limited by the structure and resolution of the initial grid. Mesh enrichment algorithms for steady problems [23, 24] generally advance the solution towards steady state on an initial coarse grid and then obtain an estimation of the error in each computational cell by using an error indicator. For the Euler or Navier-Stokes equation systems, the error indication is normally based upon a key-variable eg. the density is a popular choice for the Euler equations. Indicators based upon interpolation theory can be used, with equi-distribution of the error being the object of the refinement process [25]. The cells exhibiting largest error are automatically subdivided and the computation proceeds, with this process being repeated until the analyst is satisfied with the solution quality.

The mesh enrichment approach works well in practice [7] and Figure 4 shows the solution of problem of regular shock reflection at a wall which has been solved in this manner. This solution was produced using the basic solution algorithm described above.

The extension of the mesh enrichment concepts to the solution of transient problems has been demonstrated recently [26]. Now, as the flow features of interest are moving through the solution domain, for economy of computation mesh enrichment has to be combined with the capability of derefining the mesh in regions where the error indication is small. This work has produced a highly vectorizable algorithm, with low storage requirements, and with the ability to recover the original grid when the flow feature of interest has passed. The performance of the algorithm is shown in Figure 5 which displays the computed FCT solution for the problem of a Mach 10 shock impinging upon a half-cylinder. The solutions are depicted at three selected times during the transient.

A drawback of the mesh enrichment approach is that it provides a uniform local mesh refinement, whereas many flow features of interest are essentially one-dimensional in character. This suggests that directional refinement techniques could be computationally more efficient and work in this area has already begun. If element error indicators are replaced by indicators along element sides [27], directional refinement for steady problems can be achieved, along with derefinement. This is illustrated in Figure 6 which again shows the solution obtained for the problem of regular shock reflection at a wall using the basic solution algorithm. An alternative approach, is to use the mesh generator described earlier to regenerate the mesh based upon information provided by the computed solution on the current mesh [19]. Figure 7 shows the problem of Mach 25 flow past a blunt body at an angle of attack of 20° which has been solved using FCT in this manner with a sequence of three grids.

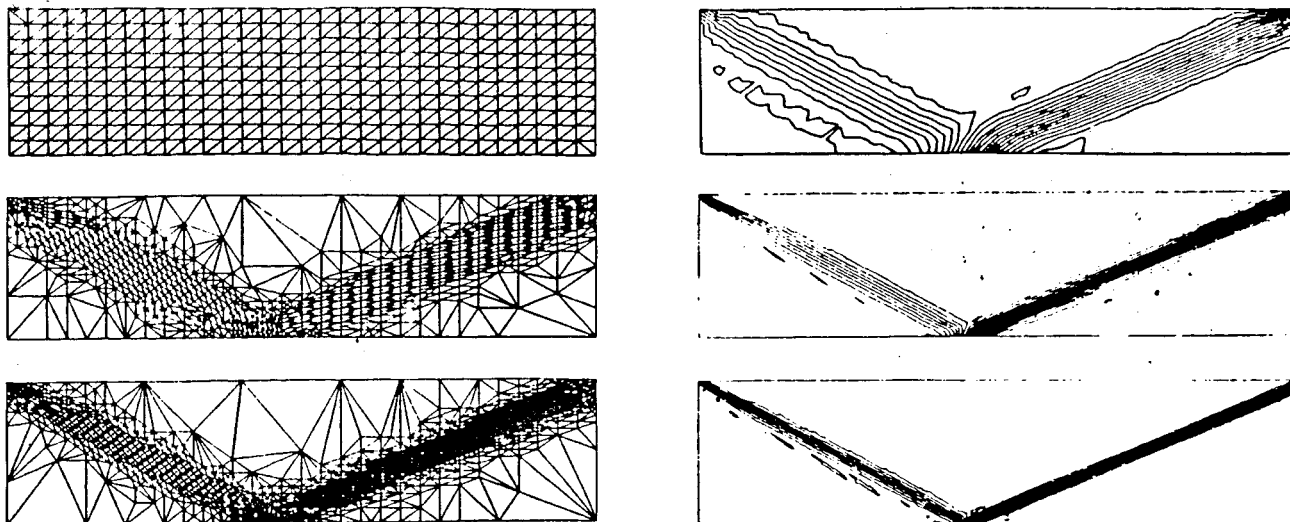


Figure 6

Regular shock reflection at a wall. Sequence of meshes produced using directional refinement and the corresponding pressure contours.

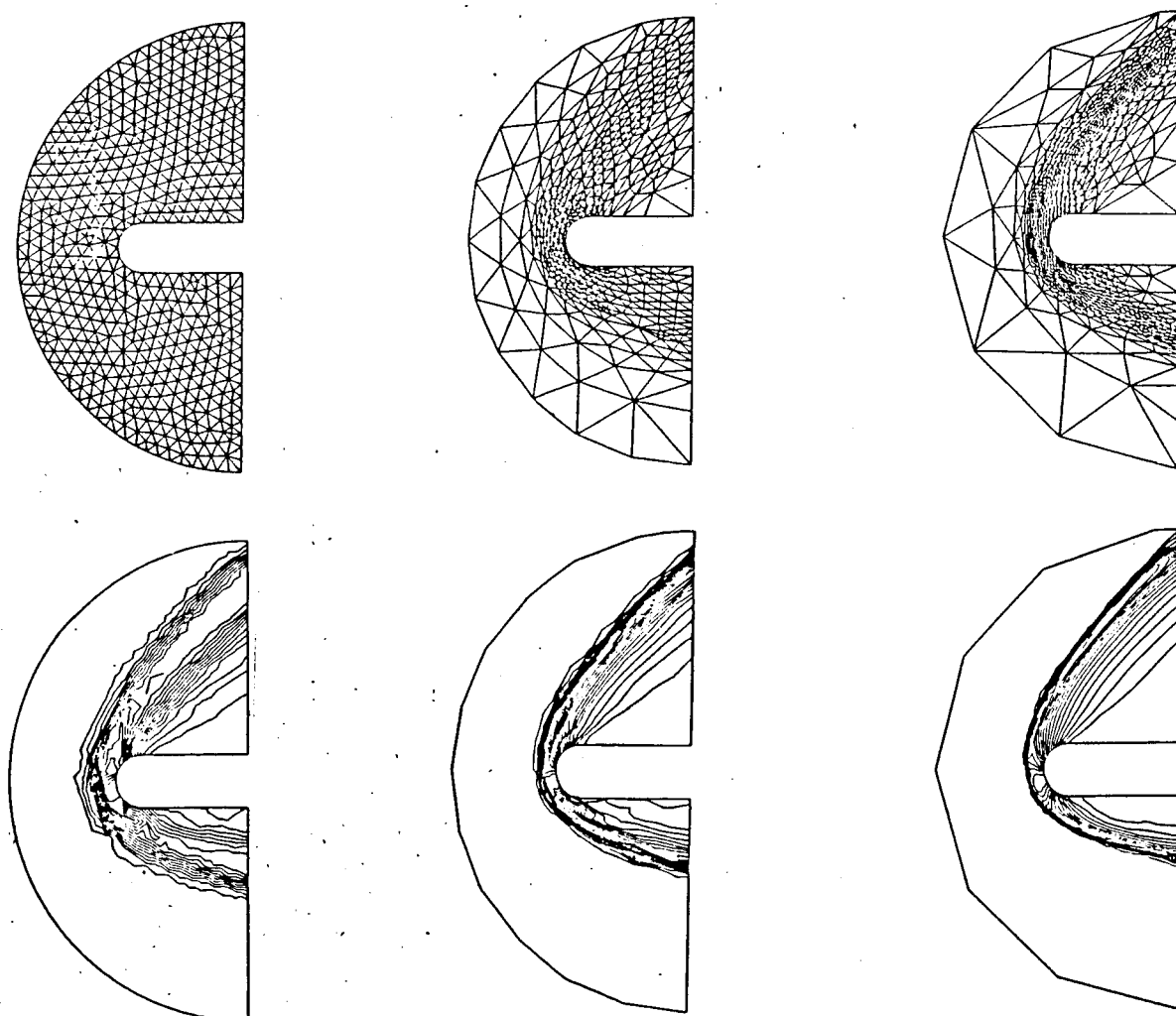


Figure 7

Mach 25 flow past a blunt body at 20° angle of attack. Sequence of meshes produced by adaptive mesh regeneration and the corresponding density contours.

ACKNOWLEDGEMENTS

The authors would like to thank the Aerothermal Loads Branch of the NASA Langley Research Center for partial support of this work under Grant No. NAGW-478 and especially A. R. Wieting and K. S. Bey for their continued interest and encouragement. We would also like to acknowledge fruitful discussions with O. C. Zienkiewicz, J. P. Boris, D. L. Book and S. T. Zalesak.

REFERENCES

1. R. Löhner, K. Morgan, J. Peraire and O. C. Zienkiewicz, "Finite Element Methods for High Speed Flows," AIAA-85-1531-CP, 1985.
2. R. Löhner, K. Morgan, J. Peraire, O. C. Zienkiewicz and L. Kong, "Finite Element Methods for Compressible Flow," in Numerical Methods for Fluid Dynamics (Edited by K. W. Morton and M. J. Baines), Oxford University Press, 1986.
3. S. T. Zalesak, "Fully Multidimensional Flux Corrected Transport Algorithm for Fluids," J. Comp. Phys., 31, 335-362, 1979.
4. J. P. Boris and D. L. Book, "Flux Corrected Transport I: SHASTA, A Fluid Transport Algorithm that Works," J. Comp. Phys., 11, 38-69, 1973.
5. J. Donea, "A Taylor-Galerkin Method for Convective Transport Problems," Int. J. Num. Meth. Engng., 20, 101-119, 1984.
6. O. C. Zienkiewicz and K. Morgan, "Finite Elements and Approximation," Wiley, New York, 1983.
7. R. Löhner, K. Morgan and O. C. Zienkiewicz, "An Adaptive Finite Element Procedure for Compressible High Speed Flows," Comp. Method Appl. Mech. Engng., 51, 441-464, 1985.
8. J. Donea and S. Giuliani, "A Simple Method to Generate High-Order Accurate Convection Operators for Explicit Schemes Based on Linear Finite Elements," Int. J. Num. Meth. Fluids, 1, 63-79, 1981.
9. R. Löhner, K. Morgan and O. C. Zienkiewicz, "The Use of Domain Splitting With An Explicit Hyperbolic Solver," Comp. Meth. Appl. Mech. Engng., 45, 313-329, 1984.
10. S. Z. Burstein, "Finite Difference Calculations of Hydrodynamic Flows Containing Discontinuities," J. Comp. Phys., 2, 198-222, 1967.
11. R. Löhner, K. Morgan and J. Peraire, "A Simple Extension to Multidimensional Problems of the Artificial Viscosity Due to Lapidus," Comm. Appl. Num. Methods 1, 141-147, 1985.
12. G. Erlebacher, "Solution Adaptive Triangular Meshes with Application to the Simulation of Plasma Equilibrium," Ph.D. Thesis, Columbia University, 1984.
13. A. K. Parrott and M. K. Christie, "FCT Applied to the 2D Finite Element Solution of Tracer Transport by Single Phase Flow in a Porous Medium," in Numerical Methods for Fluid Dynamics (Edited by K. W. Morton and M. J. Baines), Oxford University Press, 1986.

14. R. Löhner, K. Morgan, M. Vahdati, J. P. Boris and D. L. Pock, "FEM-FCT: Combining Unstructured Grids with High Resolution," University College of Swansea, Report C/R/539/86, 1986.
15. K. Morgan, R. Löhner, J. R. Jones, J. Peraire and M. Vahdati, "Finite Element FCT for the Euler and Navier-Stokes Equations," in the Proceedings of the 6th International Symposium on Finite Element Methods in Flow Problems, INRIA, Paris, 89-95, 1986.
16. R. Löhner, K. Morgan and J. Peraire, "Further Generalizations of FCT," to be Submitted to Comm. Appl. Num. Meth., 1986.
17. J. C. Cavendish, "Automatic Triangulation of Arbitrary Planar Domains for the Finite Element Method," Int. J. Num. Meth. Engng., 8, 679-696, 1974.
18. S. H. Lo, "A New Mesh Generation Scheme for Arbitrary Planar Domains," Int. J. Num. Meth. Engng., 21, 1403-1426, 1985.
19. J. Peraire, M. Vahdati, K. Morgan and O. C. Zienkiewicz, "Adaptive Remeshing for Compressible Flow Computations," University College of Swansea, Report CR/R/544/86.
20. L. Formaggia, J. Peraire and K. Morgan, "Simulation of a Store Separation Using the Finite Element Method," Submitted to Comp. Meth. Appl. Mech. Engng., 1986.
21. P. A. Gnoffo, "A Finite Volume Adaptive Grid Algorithm Applied to Planetary Entry Flowfields," AIAA J., 21, 1249-1254, 1983.
22. R. Löhner, K. Morgan and O. C. Zienkiewicz, "Adaptive Grid Refinement for the Compressible Euler Equations," in Accuracy Estimates and Adaptivity for Finite Elements (Edited by I. Babuska et al), J. Wiley and Sons, 1986.
23. J. F. Dannenhoffer and J. R. Baron, "Grid Adaptation for the 2D Euler Equations," AIAA Paper 85-0484, 1985.
24. F. Angrand, V. Billey, A. Dervieux, J. Periaux, C. Pouletty and B. Stoufflet, "2D and 3D Euler Flow Calculations with Second Order Accurate Galerkin Finite Element Method," AIAA Paper 85-1706, 1985.
25. J. T. Oden, "Notes on Grid Optimization and Adaptive Methods for Finite Elements," University of Texas at Austin, TICOM Report, 1983.
26. R. Löhner, "An Adaptive Finite Element Scheme for Transient Problems in CFD," Submitted to Comp. Meth. Appl. Mech. Engng., 1986.
27. R. Löhner and K. Morgan, "Improved Adaptive Refinement Strategies for Finite Element Aerodynamic Computations," AIAA-86-0499, 1986.